ZHEJIANG FLASHFORGE 3D TECHNOLOGY CO., LTD.

Creator Pro Start-up Guide

Contents

⚠️ **WARNING**

Flashforge sets the power supply to default 230v before leaving factory. If voltage in your location is between 90v to 132v, please switch the power setting from 230v to 115v before you plug in the power. **Failure to do so will damage the motherboard**. You can find the power source at the bottom of Creator Pro. The red box is where the power setting is located:

PRECAUTIONS:

Please make sure to read this page carefully prior to setting up and operating the Creator Pro.

⚠️ The Creator Pro is very sensitive to static electricity, so please make sure you contact a grounded object before operating the machine.

⚠️ Before repairing or making any alterations to the Creator Pro, it is essential that the machine is turned off and the power cord is unplugged.

⚠️ The Creator Pro operates at very high temperature; allow the nozzle, the extruded plastic and heating plate to cool before touching.

⚠️ Some plastic filaments may give off a little odor when heated. Because of this, the machine should always operate in a well-ventilated area.

⚠️ Do not wear gloves when operating or repairing, as entanglement may occur and cause injury.

⚠️ Do not leave the machine unattended when in operation.

1 What's Included in the Box?

Along with your Creator Pro 3D printer, the box also contains the following:

In the two long boxes on the top of the package, you'll find:
- Power cord x 1pc,
- USB A to B cable x 1pc,
- Sensor Line x 1pc,
- Filament guide tube x 2 pcs.
- Filament spool holder x 2 pcs.

In the Creator Pro Acrylic Accessory box:

- Acrylic sheet x6 pc
- Handle x1 pc
- Accessory bag x1 pcs

An accessory box on the top of the Creator Pro:
- Dual extruder heads x 1pc,
- 4G SD card x 1 pc,
- Blue tape x 1 pc,
- Bolt, hex wrench and screws kit x 1 bag.

Under Creator Pro's build platform, there are 2 rolls filament:
- White ABS filament x 1kg (2.2pounds),
- Blue PLA filament x 1kg (2.2pounds),

2 Un-boxing

The Creator Pro was carefully packaged at FLASHFORGE manufacturing facility. Please follow the unboxing steps laid out below.

CAUTION

⚠️ Handle the package and its contents with extra care; do not use any unnecessary force.

⚠️ Do not remove the **thin yellow film** from the heating plate. It is heat resistant tape that improves the adhesion of the extruded plastic to the plate.

⚠️ Do not remove the wrapping around the nozzle. It consists of a ceramic fiber fabric and heat resistant tape that helps to keep the nozzle at a constant temperature.

First, put the box on the floor in a clean and flat surface. Remove the top two long boxes and set them aside. You will find **a power cord, 1 USB A to B cable, 1 sensor line,2 filament guide tubes, and 2 spool holders** in these two long boxes.

You will find another box within the relevant assembly tools under the two long boxes. You should assemble the Acrylic cover yourself and please check the assembly video in the SD card.


Please hold both sides of the printer and take it out after you take out the side EPE of Creator Pro. Please make sure you only can hold the frame.


Now you can see the top of the printer along with more boxes inside. The large box with the black wire is the **accessory box.** This contains the **dual extruder heads, SD card and other important components**. Don't remove the accessory box or its contents yet. *Note:* **Do not lift the box by the black cable! If so could cause damage.**

Next, take the Creator Pro out of the box by grasping the outer frame. **Be sure to grasp only the frame.** Gently lift and transfer the printer to your work surface as shown on next page.

Gently lay down the printer and open the bubble film as shown below.
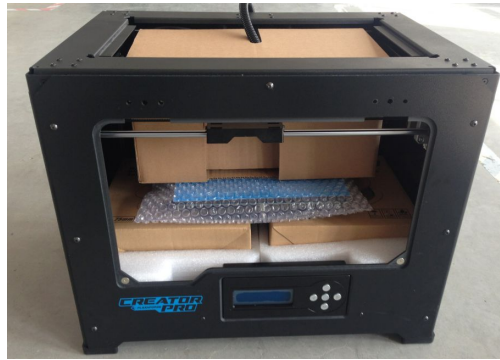
Now, open the accessory box and remove the accessory sleeve. You will find the **dual extruder heads** in the protective packaging along with the black cable, carefully remove it and place it on your work surface. Remove the cardboard packing material and take the **accessory box** from the printer, set it aside for later.
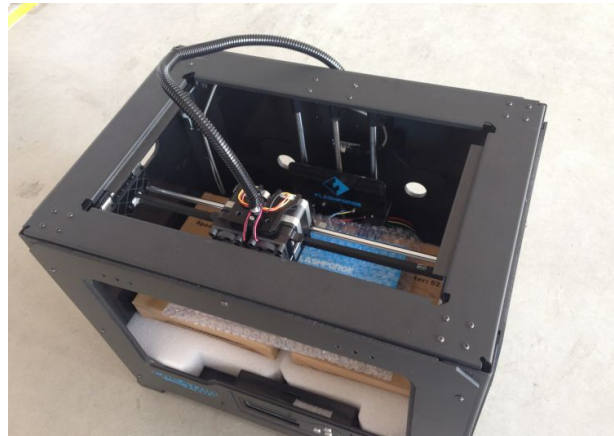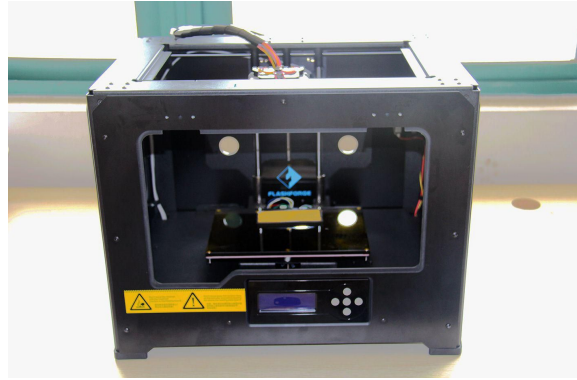

The build platform should now be visible. It is an aluminum plate covered in a thin polyamide film. This is the surface that your objects will be printed on. ***Note:*** **Remember not remove the film.**

The next step is to raise the build platform; there are two ways to do this:

1. Turn the screw which is behind the rotating platform.
2. Grasp the printing platform with one hand on each side, raising it slowly and keeping it level. Stop once the platform is just shy of the bronze nozzle.

Now you can see two rolls filaments we offer for free, you can get them easily after removeing the rest packaging.
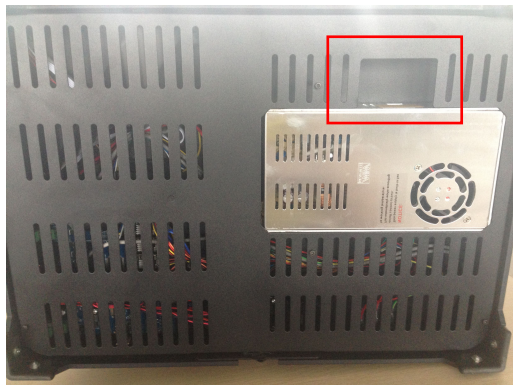
3 Initial Hardware Installation

To do this, begin by carefully placing the Creator Pro on one of its sides so that the bottom is exposed. You will see the Creator Pro's power supply. Using a flathead screwdriver switch the red power supply switch so that it displays 115v, as shown below. Now you are ready to install the extruder.

You'll need the two shortest silver screws from the kit found in the accessory box, and the appropriate hex wrench.

First, lower the build platform by using one of the methods described in the previous section. Holding the extruder by both sides, take it out of the accessory sleeve and position it on the extruder seat with the fan facing forward. Align the screw holes and fasten with the two short silver screws.
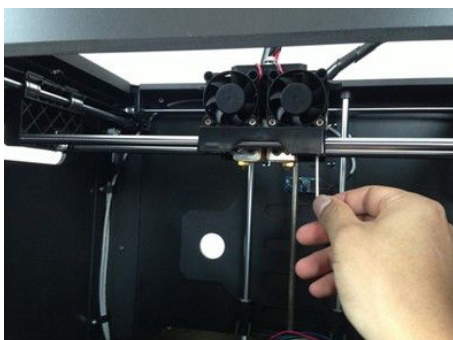
Next is the installation of the spool holder. Install one on each side.

The installation of the spool holder is very simple – just insert it into the circular opening and tighten the nut behind. Then install the **filament guide tube** to the empty spot on the extruder. Place one end of the guide tube into the hole. The hardware installation is almost complete.
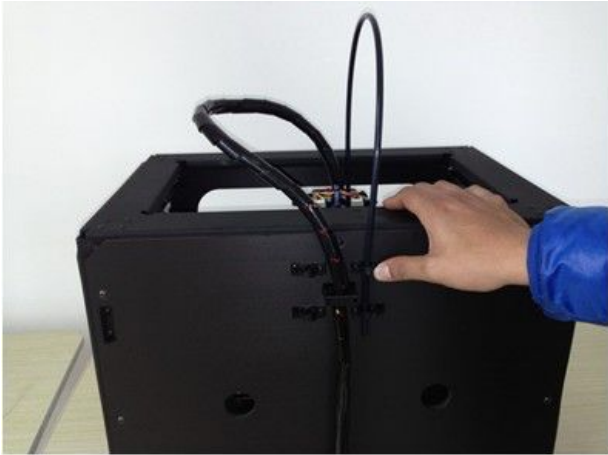
Next, with the power switch in the **OFF** position, confirm that the power cord is plugged into the power outlet next to the power switch.

Now plug the **USB A to B cable** into the USB B-type port, do not plug the other end in yet.

Finally install the filament on the **spool holder** and screw in the nut. **Do not over tighten.**

Congratulations! You have completed the initial hardware installation! If you're ready to start printing, proceed to the next step: Software Installation.

4 Software Installation

> **Note** **SPECIAL NOTES TO WIN 8 USERS BEFORE SOFTWARE INSTALLATION**
>
> Win 8 users please make sure that you have disabled the Driver Signature Enforcement setting before installing the ReplicatorG software. Below is a video link showing you how to accomplish this:
>
> http://www.youtube.com/watch?v=NM1MN8QZhnk

ReplicatorG0040 is the ideal software to use with the dual extruder Creator Pro. It can be downloaded from our official website by following the link below and click on Download.

http://replicat.org/download

ReplicatorG0040 for Windows users can also be found in the folder of **information of 3D printer** on the SD card, which shipped together with the Creator Pro.

Steps for installing your software:

1. Download the appropriate version of ReplicatorG suitable for your system from http://replicat.org/download.
2. From the SD card shipped with your printer, browse to the "information of 3D printer" folder and run the Python installation file and the Python acceleration components.
3. After completing the Python component installation, click Replicator-0040-Installer to install the ReplicatorG software.
4. To run the ReplicatorG software, double click the shortcut on your desktop or in your start menu.
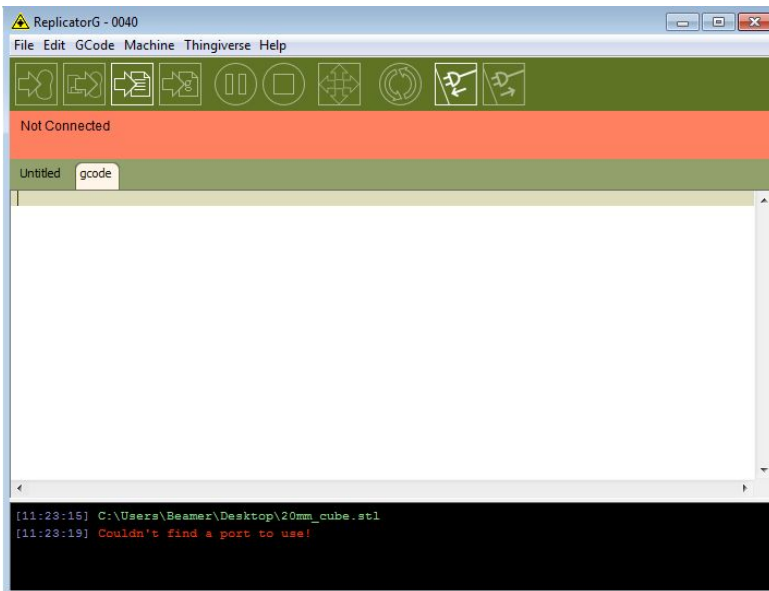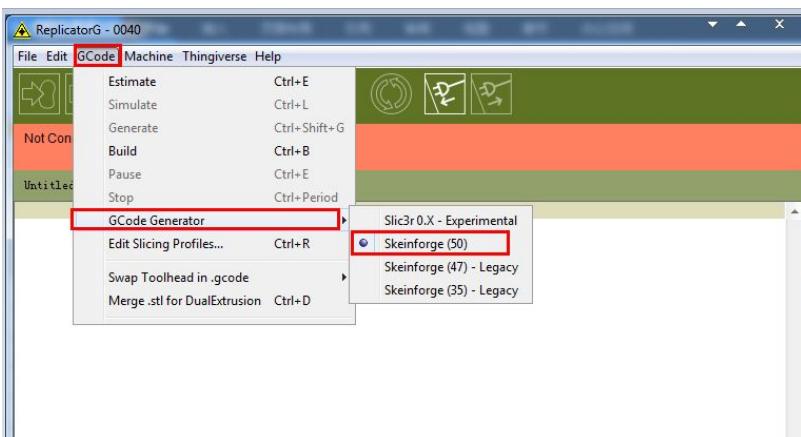
After opening ReplicatorG software, click on **Machine** to select your **Machine Type.** Choose **The Replicator Dual** for Creator Pro dual extruder machine.
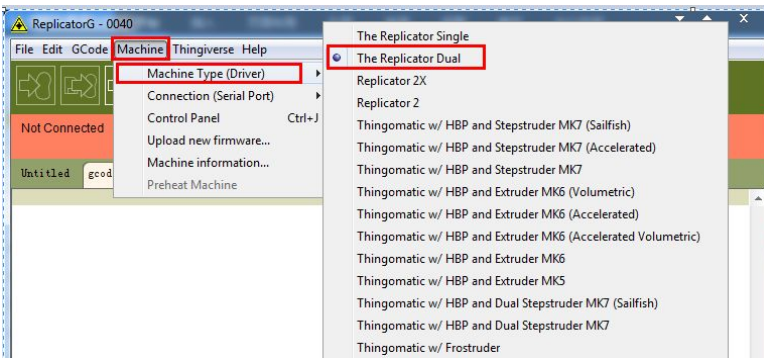

After selecting the appropriate machine type, click on **GCode** on the top navigation bar, and under **GCode Generator**, select **Skeinforge(50)**.

The following gives an introduction on how to import files (.STL) into the Replicator G software and then generate Gcode to print your creation.

Click **File** > **Open**, then browse and select the file (.STL) that you would like to print. Import the file by double-clicking on it. Then the drawing will appear on the Replicator G interface.
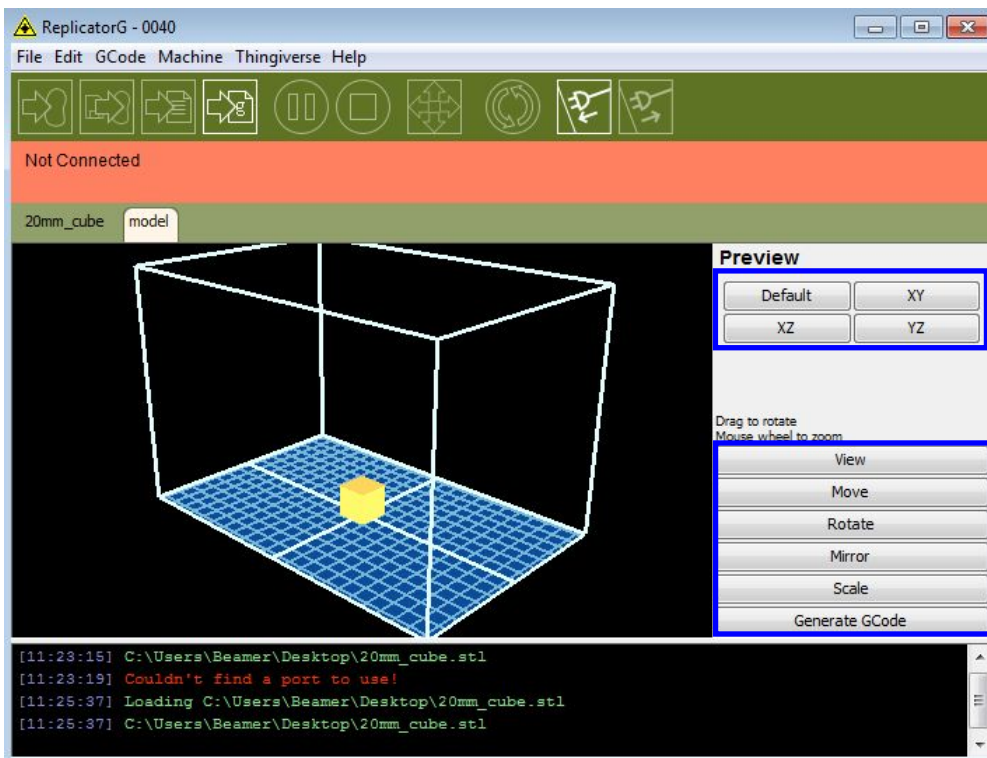
When the object is imported you may find that it is not on the virtual build platform or even on the screen, using the function keys indicated by the blue boxes in the illustration above, you can change the camera angle and reposition the object onto the center of the build platform. Once you've done this, the next step is to generate the Gcode. This is achieved by clicking on the button **(Generate GCode)** at the bottom of the panel.

Note **EXPLANATION OF GCODE SETTINGS**

A new window will pop up, giving you several options on how the Gcode will be generated:
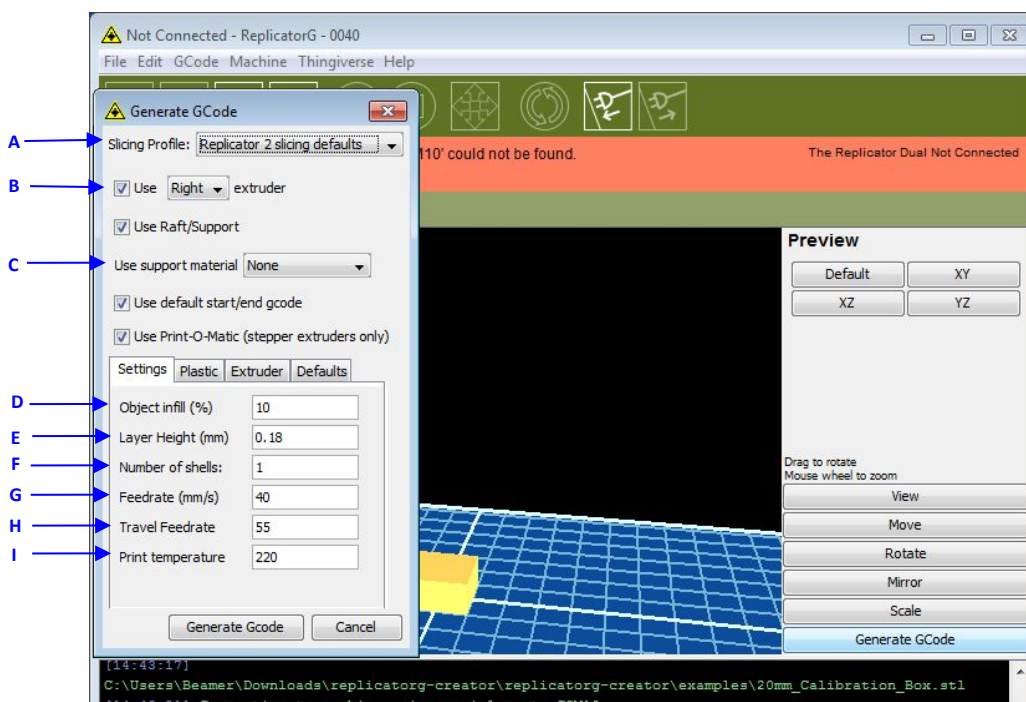
A. **Slicing Profile:** select **Replicator slicing defaults for ABS printing, then select Replicator 2 slicing defaults for PLA printing.**

B. This tells the printer which extruder to use for a dual-extruder head printer, either the **left** or **right** head can be selected.

C. If your sample will have any hanging surfaces, it is recommended to have support. Select **None** indicates that there is no support. **Exterior** means surface support. **Full support** means all support.

D. **Object infill:** where **100%** is a solid print, **0%** is a hollow object. The recommended setting is **10%;** this will save time and filament. Low infill also can reduce the corners lift up issue during ABS printing.

E. **Layer Height:** this controls the vertical resolution of the print. The recommended thickness is **0.27mm**.

F. **Number of shells:** this is the wall thickness; it's usually set at **1**.

G. **Feedrate** is the speed at which the filament is fed into the extruder, and this is usually set between **30** and **100**. For **ABS printing**, **60** is recommended; for **PLA printing**, **100** is recommended.

H. **Travel feedrate** is the speed at which the printer head moves over the base, and it's usually set between **30** and **120**. For **ABS printing, 80** is recommended; for **PLA printing, 120** is recommended.

I. **Print Temperature** is the temperature to which the nozzle is heated. This varies between filament types. Set it to 220 for the default filament.

After you finished inputting in the settings, click **Generate Gcode** and the progress bar will appear.
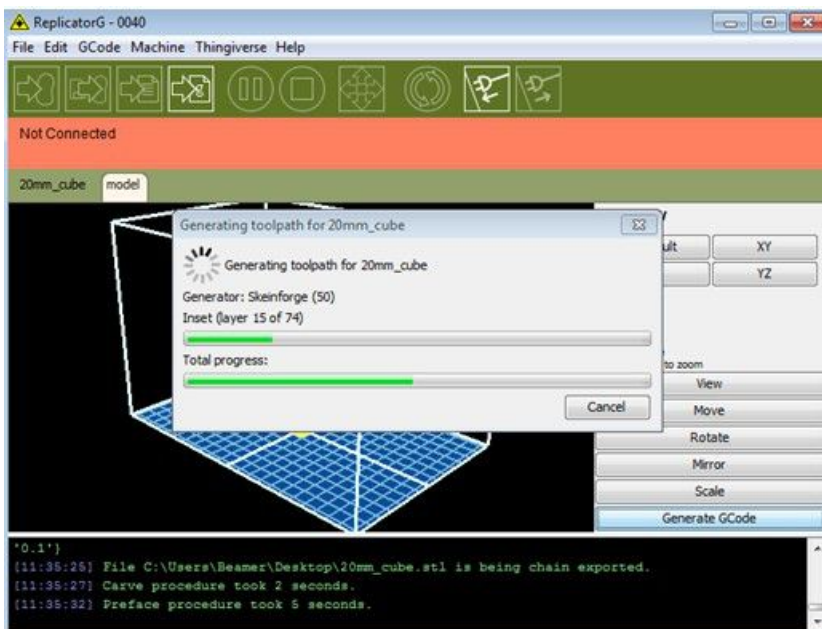
For users who did not choose the default installation path in the installation of Python, clicking the **Generate Gcode** button in Replicator G will result in dialogue box popping up alerting that the executable Python file cannot be found.

In order to solve this, click the **No** button to dismiss the dialogue box. Next, we need to configure the corresponding menu.
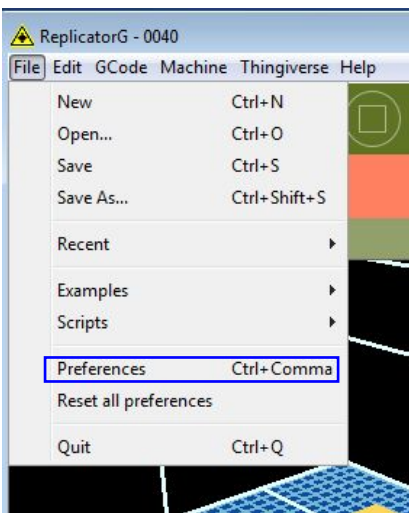
Click **File** > **Preferences**, then click **Select Python interpreter** under the **Advanced tab**. A window will pop up. Navigate to the Python installation directory, and select **python.exe** and click **Open**.
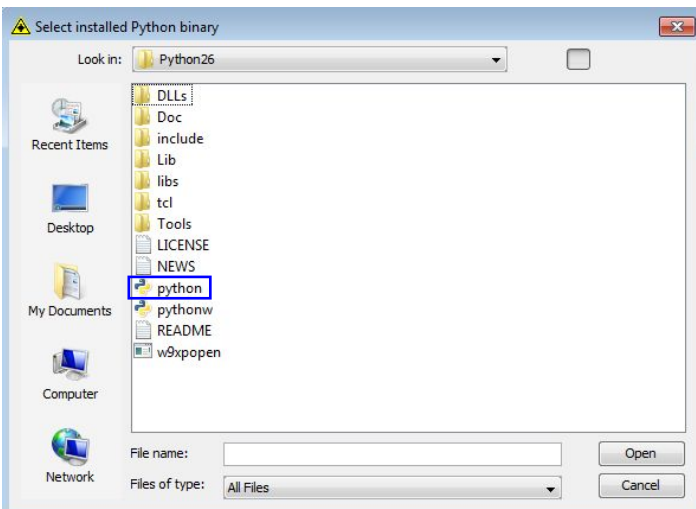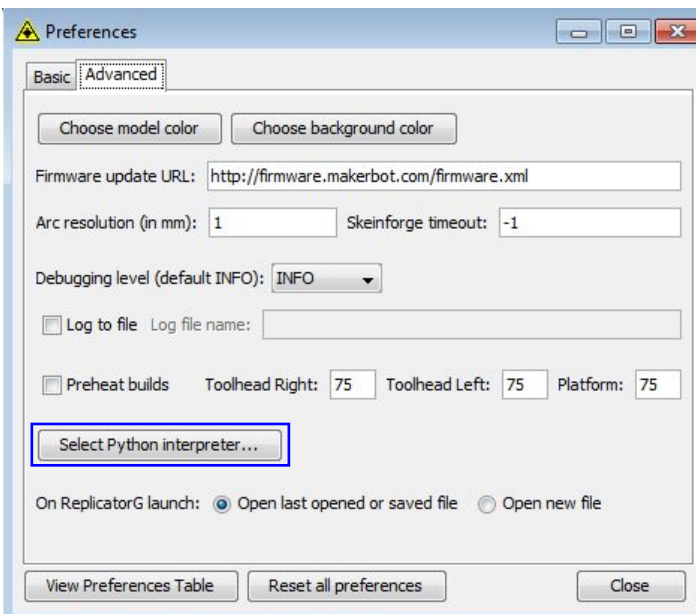

Last, click **Close** on the Preferences menu and you are done!

The machine will now work as normal when generating Gcode, and the Python error message will not pop up again.In the next section, we will start a preliminary test on the machine's connections, and heat the platform and extruder so that the extruder is ready for its first print.

5 USB Connection & Temperature Setting

First, connect the machine and computer with the provided **USB A to B cable**. The USB port on the machine is bound by the box in the below image. After connecting the cable, open the ReplicatorG software as we are going to link the computer and printer.
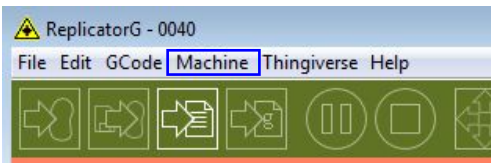

Inside the ReplicatorG software, click **Machine > Connection (serial port) > Rescan serial ports**, if no new ports appear then the software driver has not been installed. To install the driver manually, go to **My computer** and right click on **Properties.** The basic system parameters appear. Next select **Device Manager**.
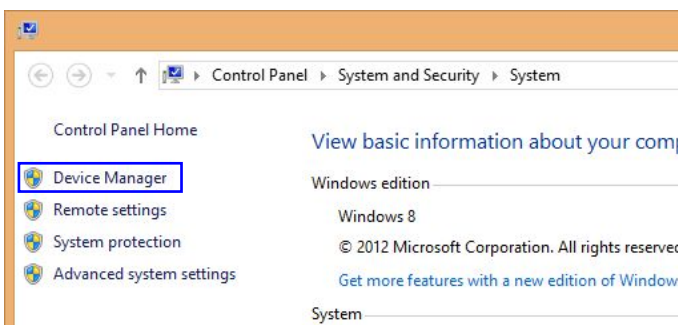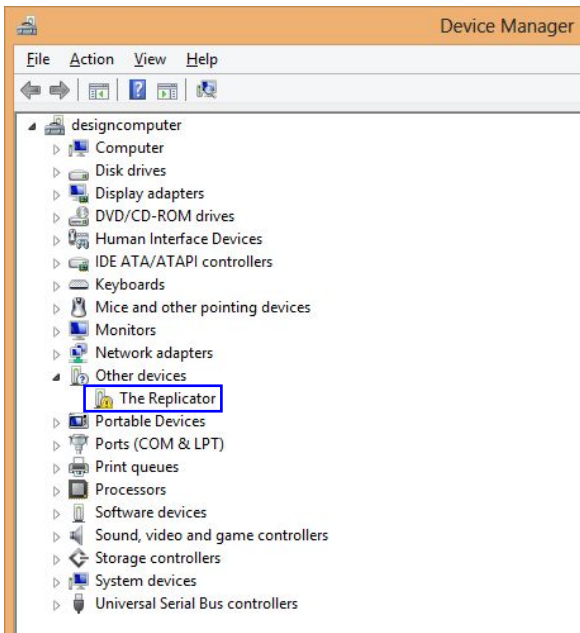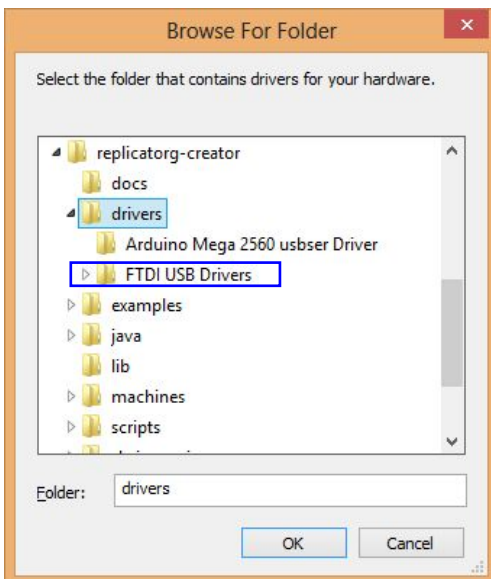
Locate the software driver shown in the blue box above. Right click and select **Update Driver Software**.
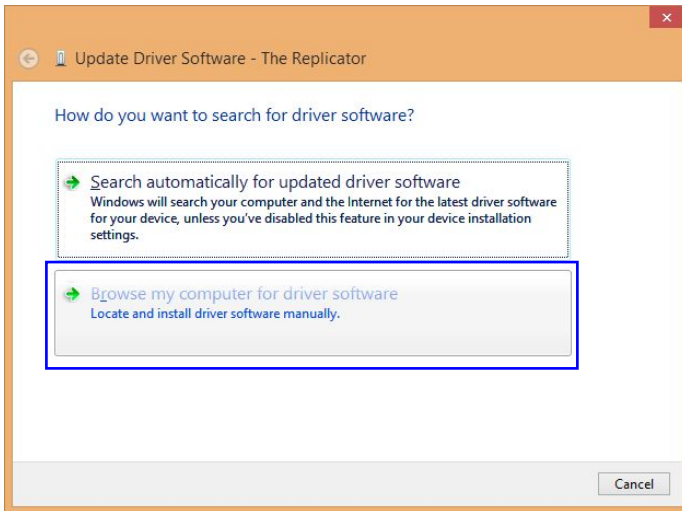

Click **Browse my computer for driver software** to find the location of ReplicatorG0040 on your system.

Click **FTDI USB Drivers** in the driver folder before confirmation and click **OK**. The drivers will then be installed.

The next step is to link the printer. Rescan the serial ports and select the one that appears on your machine **(on our test machine the port was COM10, but different computers vary)**.
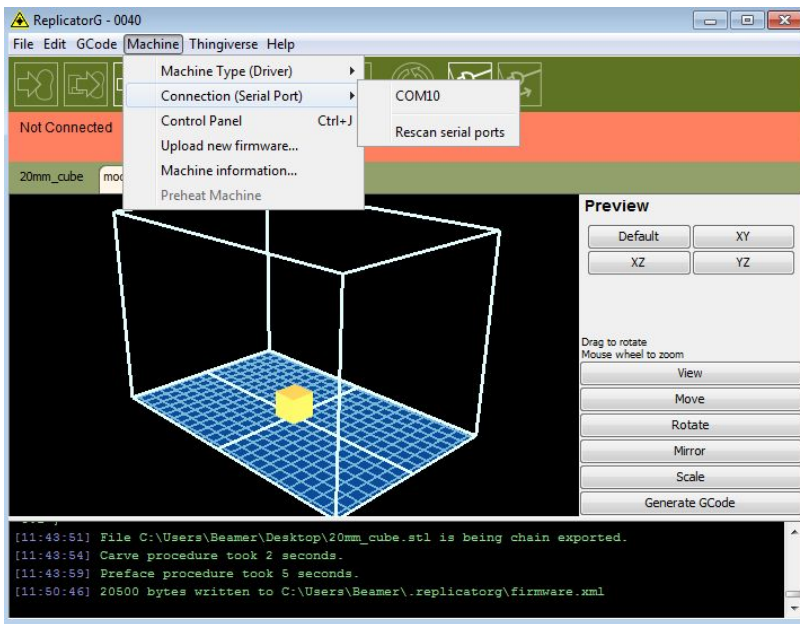

Now we can link the machine to the computer. Click on the printer connect icon highlighted in blue :

If the printer connect icon turns dim, as shown in the illustration below, the printer has successfully connected to the computer. Next we need to preheat the extruder and build platform.
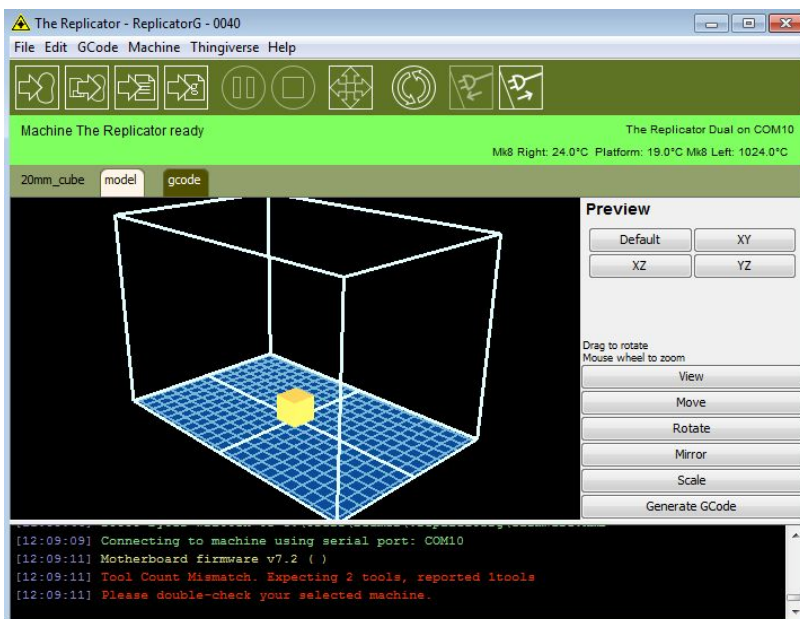
In the same set of menu, click on the cross-shaped icon and a new interface will appear.
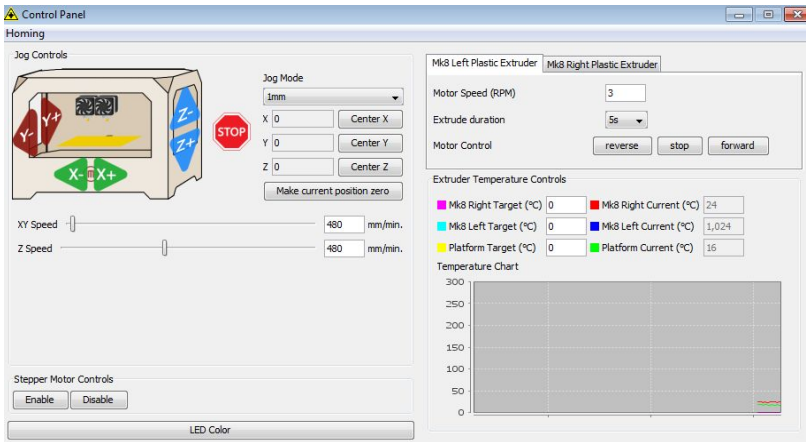
Input the following target values: 220°C for the extruder and 115°C for the heating platform. After entering the values, the platform will start to warm up. When the extruder temperature reaches 50°C, the cooling fan will be activated and the current temperature value will display to the right as shown below.
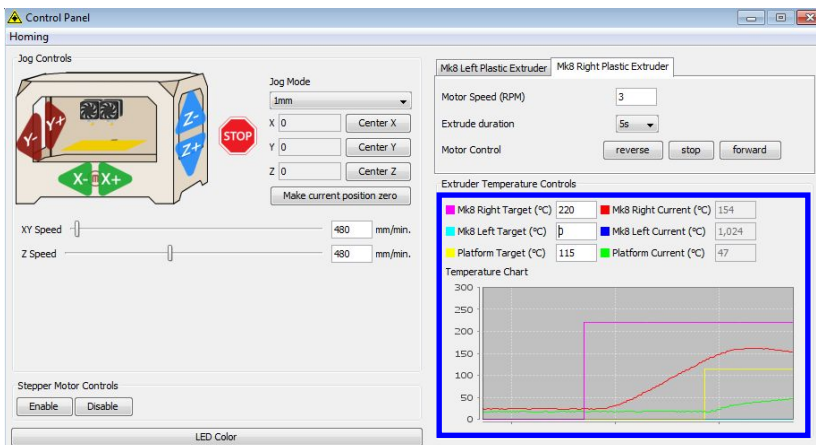
6 Filament

To make the process of feeding or withdrawing the filament easy, please follow the next few steps carefully:

After inserting the filament into the feeding hole, do not push it further until the extruder temperature reaches 200°C or more. Once the machine reaches this point, you will feel the filament being pulled into the extruder head.

6.1 Install Filament

First, remove the filament guide tube from the extruder head.
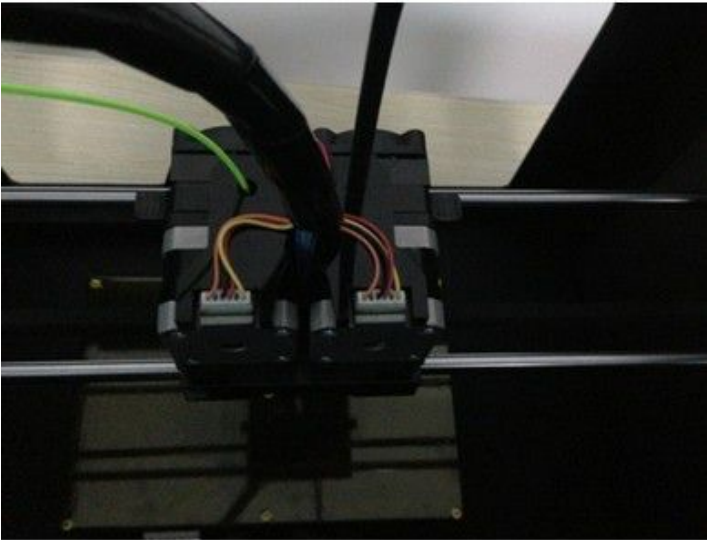
When you have removed the guide tube you can remove the filament that is inside the guide tube. To avoid any blockages during printing, please ensure that the two threads are loaded from the middle. There are two wire trays, one runs clockwise and the other one runs counter - clockwise, as shown below:

There are two ways of feeding the filament into the extruder head, one is using the LCD screen on the printer itself and the other is by using the control panel on the Replicator G software.

6.2 Feeding the Filament Using the LCD Screen

1. Turn on the Creator Pro; the display will indicate:
   ▶ Print from SD
   Preheat
   Utilities

2. Using the directional arrows to the right of the screen, press the page down key to scroll to the next page. The display will show:
   Preheat
   ▶ Utilities
   Info and Settings

3. Select **Utilities**; press the OK key of the keypad. The display will show:
   Monitor Mode
   ▶ Filament loading
   Preheat Setting
   Level Build Plate

Feed operation with LCD screen

4. Select **Filament loading**; press the **OK key** of the keypad. The display will show:
   ▶ Load Right
   Unload Right
   Load Left
   Unload Left

5. Select the appropriate side of the extruder you wish to load(Left or Right for models with dual extruders). Press the **OK key** again on the keypad. The display will indicate: I'm heating up my extruder!" At this time, the temperature of the right nozzle is being heated up. When the temperature of the nozzle reaches its target temperature, pressing the **OK key** on the keypad, and the nozzle should start extruding material. If not, keep pressing the **OK key** unit it does.

6.3 Withdrawing the Filament Using LCD Screen

CAUTION

⚠️ If you just finished your printing or already extruded the filament and want to withdraw the filament, at this moment, the extruder is still over 200°C, first push the filament in a little further, and then directly pull it out.

⚠️ If you want to change another color filament, first you need to withdraw filament and then load in. Please do as following shown to avoid filament jam.

1) Turn on the Creator Pro, the display will indicate:
   Build from SD
   ▶ Preheat
   Utilities

2) Using the directional arrows to the right of the screen, press **page down key** to scroll to the next page and select Preheat, Press **OK key** on the keypad, then you will see：
   ▶ Start Preheat
   Right Tool   OFF
   Left Tool   OFF
   Platform   OFF

3) Press **page down key** to select **Left Tool (or Right Tool)**, click **OK key**, you will see：
   ▶Start Preheat
   Right Tool   ON
   Left Tool   ON
   Platform   OFF

4) Press **page up key** back to Start Preheat, and Press **OK key**, you will see:
   Heating:
   R Extruder: 033/230C
   L Extruder: 033/230C
   Platform: 024

This means the left extruder is heating up, when it reaches 220°C. First **push in** the filament **a little bit until you see filament come out of the nozzle,** then **pull it out quickly**. This will ensure you have withdrawn the filament inside the nozzle successfully.

6.4 Feeding the Filament Using Replicator G Control Panel

The following can only be performed while the printer is connected to the computer.

Open the Replicator G software, click the icon in the red box, this is the control panel icon. The dialogue box shown below will then pop up.
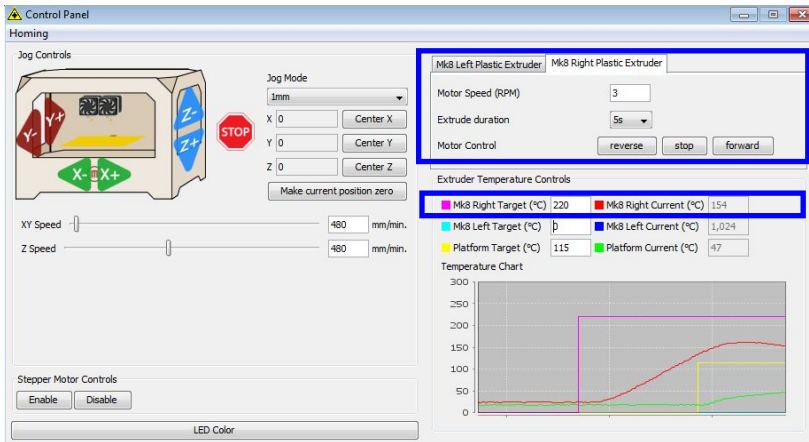
To heat the right extruder and feed material, click the right extruder on the upper right corner of the control panel and manually modify the temperature in the "Right Target" setting for the extruder temperature control changing the temperature to 220°C. A red line will be plotted in the temperature map. When the actual (Current) temperature reaches 220°C you can pull the filament out.

6.5 Setting Parameters

Before generating Gcode, specific parameters can be set up to customize various aspects of the print. An explanation to each of the settings can be found in the **EXPLANATION OF GCODE SETTINGS**.

7 Initial Print

CAUTION

⚠️ Creator Pro has been leveled perfectly before leaving factory. However, it will get unleveled during long distance

shipping, please do level the build plate again before starting a print. See below in instruction.

First, make sure that you've completed all the steps in the **Un-boxing and Hardware Setup section**.

1. Your dual extruder heads should be bolted in place,
2. Your filament guide tubes is connected,
3. Your filament is mounted on the spool holders.

Once all these have been completed, you can plug in your power supply. If everything is ready, then turn on the power switch on the back of your Creator Pro.

Every printer will be leveled before shipped out, but we can't ensure that the platform won't move during delivery, so it will be better to level platform before you begin to print. We will put one SD card within the 3D printer，there is a file named **PlateLeveling.x3g.**Put the SD card into the slot,choose **Print from SD ,**click **OK,**then choose **PlateLeveling.x3g**and click it, you will see the content below:**Find the 3 knobs on the bottom of the platform and tighten four or five turns. I'm going to move the extruder to various positions for adjustment.In each position, we will need to adjust 2 knobs at the same time. Nozzles are at the right height when you can just slide a sheet of paper between the nozzle and the platform. Grab a sheet of paper to assist us. Please wait.You will find 3 knobs under the platform,the extruder will move to different locations, in everylocation, you need to rotate2 knobs toadjust.You can put one piece of paper between nozzle and platform,ifwhen you move the paper and can feel thefriction(which means you can move the paper but it can't be scratched), then level finished**
During the process, what you should do is only click OK continually, the screen will go to next page; and you also only

need to click OK, the extruder will move to next point.

Attention please: when the 2 knobs level finished, you only need to click **Print Another Copy, then the extruder will continue to the third point, four points can be level finished**
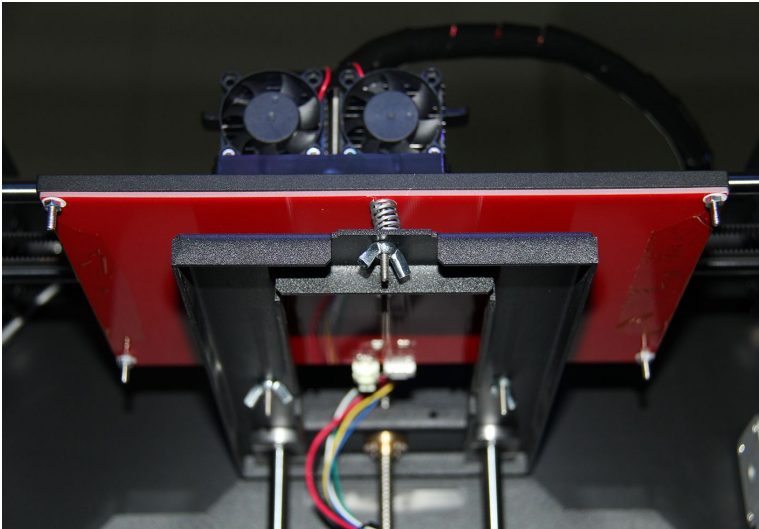

Now, let's start the **initial print**. Click **File** > **Examples** and then select **the 20mm_Calibration_Box.stl.** The preview interface will then appear along with a virtual impression of the 20mm cube on the virtual printer bed. On the preview interface click **Move** > **Center** and **Put on platform** so that the sample will be printed on the center of the build plate.
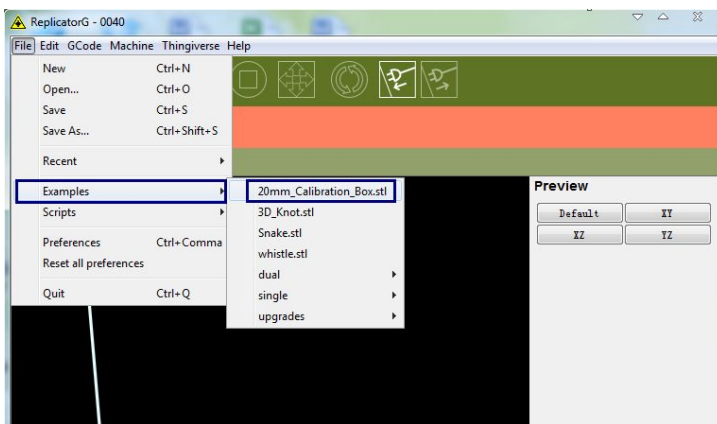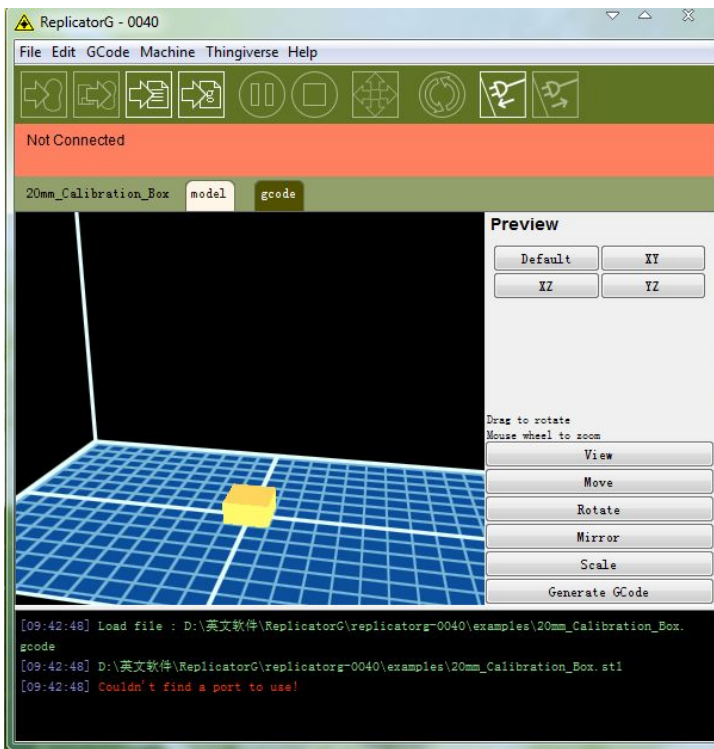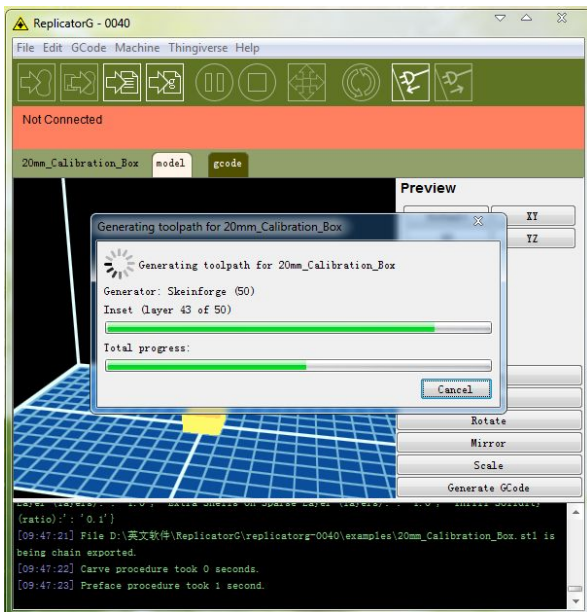
Next click **Generate GCode**, a dialogue box will pop up, after inputting your chosen parameters the Gcode will be generated. Please refer to **EXPLANATION OF GCODE SETTINGS.**
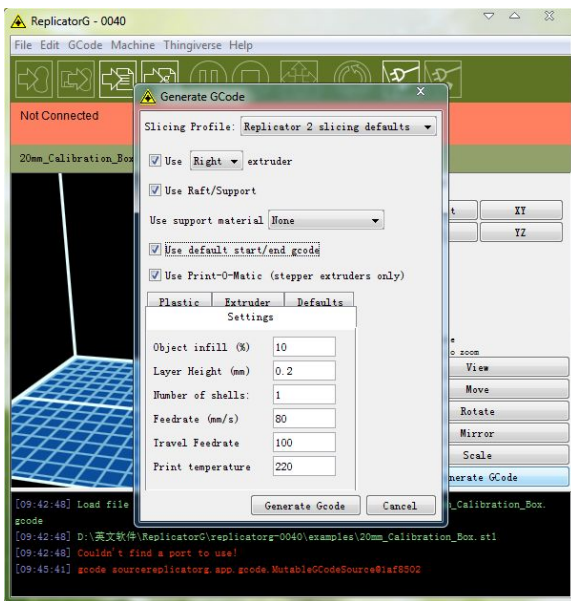

A loading bar will then appear showing the progress of the Gcode generation.

After the Gcode is generated, the temperature of the bottom plate needs to be changed if you print PLA filament. See below for the required modification. First, select Gcode as shown highlighted in the illustration below and change **M109 S110** to **M109 S50** to set the HBP temperature at 50°C. Then click **File** > **Save** to save this modification. If you print ABS filament, then no HBP temperature modification is needed.
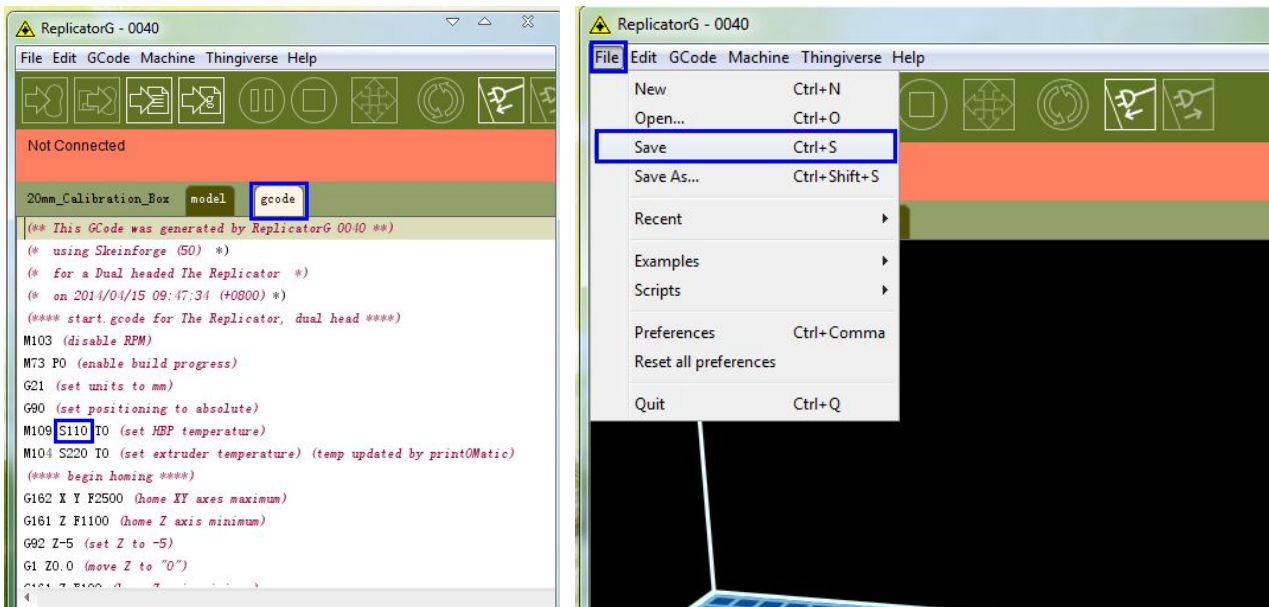

If you have connected Replicator G Creator Pro printer, click the left most button on the green bar near the top of the screen. The Gcode has now been sent to the printer and the object will start printing shortly. If not, please refer to **USB Connection & Temperature Setting** to connect.
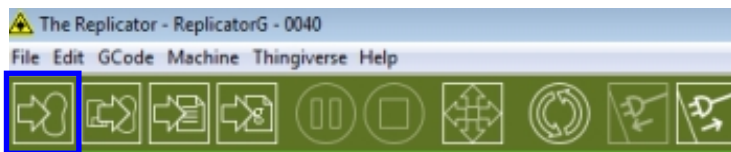
8 Dual Extruder Print

Open ReplicatorG and select **Gcode** > **Merge .stl for Dual Extrusion**. You will see following:
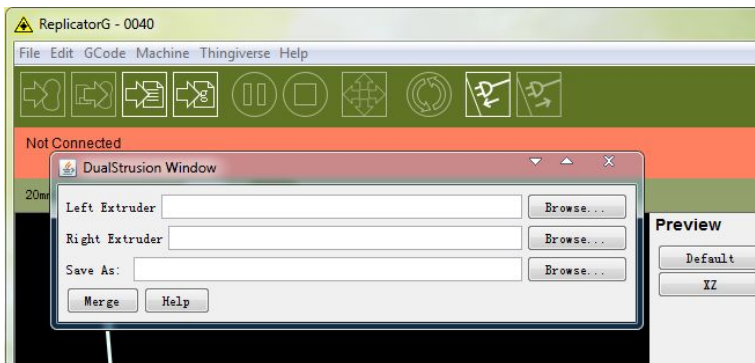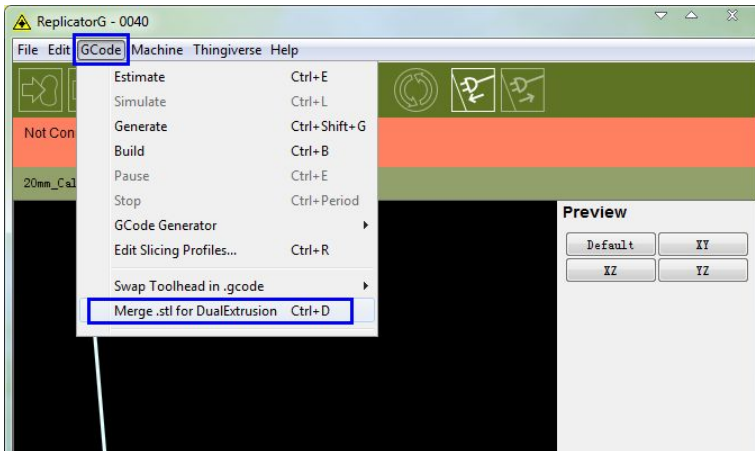

Click on the first **Browse** button to locate and select the file for left extruder; and then click on the second **Browse** button for right extruder. For example, you can find the **examples** folder in its installation path. Open the folder **examples** select **Two_color_world_a.stl for** left extruder and repeat the process for the right extruder as shown below:
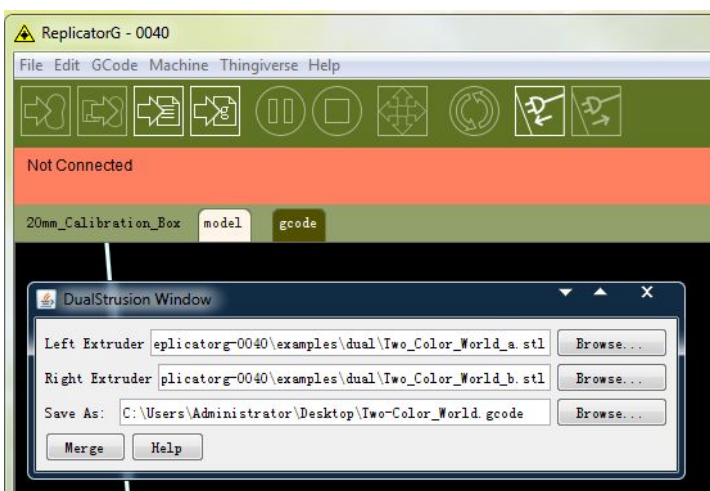

Now you need to save it to the desktop with the suffix format .gcode. Click **Merge**, two dialogue boxes will then pop up.

NOTE: Do not check "the 'use raft/support' option. If you want to print dual colors with PLA filament, then perform the following settings:
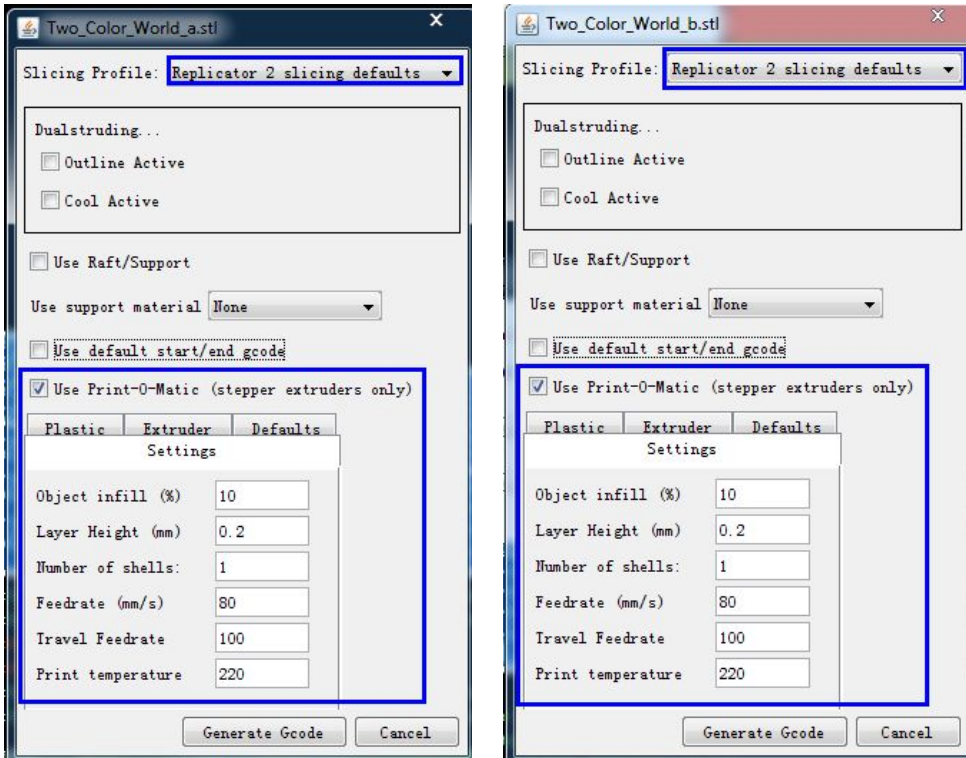
If you want to print dual colors with **ABS filament**, select **Replicator slicing defaults** under the **Slicing Profile** column, and lower the **Feedrate** to **60** and **Travel Feedrate** to **80**.

Click the **Generate Gcode** option for both boxes. After the Gcode is generated, click on the **gcode** tab as highlighted below and change HBP temperature to **50** if you are printing with PLA and save this modification by going to **File** > **Save**.
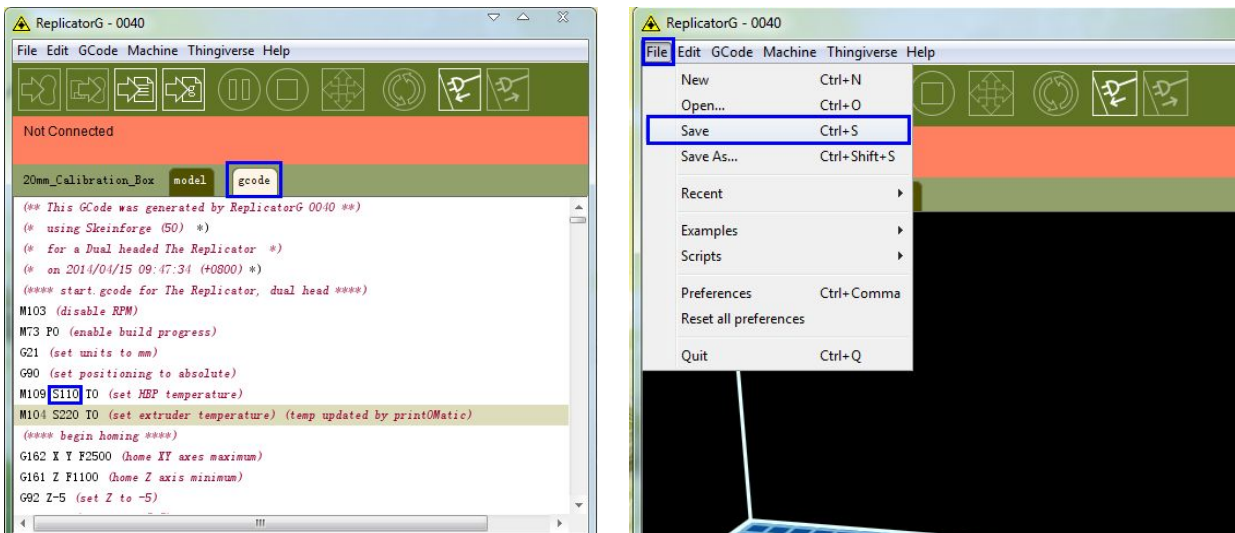
Click on the following button to export **.x3g** file for SD card print.